Naval Surface Warfare Center

Carderock Division

West Bethesda, MD 20817-5700

NSWCCD-61-TR-2015/31 **November 2015**

Survivability, Structures, Materials, and Environmental Department

Technical Memorandum

Simulation of Weld Mechanical Behavior to Include Welding-Induced Residual Stress and Distortion: Coupling of SYSWELD and Abaqus Codes

by

Charles R. Fisher, Ph.D.

Ken Nahshon, Ph.D.

Simulation of Weld Mechanical Behavior to Include Welding-Induced Residual Stress and Distortion:
Coupling of SYSWELD and Abaqus Codes

NSWCCD-61-TR–2015/31

Naval Surface Warfare Center

Carderock Division

West Bethesda, MD 20817-5700

NSWCCD-61-TR-2015/31 **November 2015**

Survivability, Structures, Materials, and Environmental Department

Technical Memorandum

Simulation of Weld Mechanical Behavior to Include Welding-Induced Residual Stress and Distortion: Coupling of SYSWELD and Abaqus Codes

by

Charles R. Fisher, Ph.D.

Ken Nahshon, Ph.D.

REPORT DOCUMENTATION PAGE	*Form Approved* *OMB No. 0704-0188*

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing this collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. **PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.**

1. REPORT DATE *(DD-MM-YYYY)* 05-11-2015	**2. REPORT TYPE** Technical Report	**3. DATES COVERED** *(From - To)* Dec 2013 – July 2015

4. TITLE AND SUBTITLE Simulation of Weld Mechanical Behavior to Include Welding-Induced Residual Stress and Distortion: Coupling of SYSWELD and Abaqus Codes	**5a. CONTRACT NUMBER** N/A **5b. GRANT NUMBER** N/A **5c. PROGRAM ELEMENT NUMBER** N/A
6. AUTHOR(S) Charles R. Fisher, Ph.D. Ken Nahshon, Ph.D.	**5d. PROJECT NUMBER** N/A **5e. TASK NUMBER** N/A **5f. WORK UNIT NUMBER** N/A
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) AND ADDRESS(ES) Naval Surface Warfare Center, Carderock Division 9500 MacArthur Boulevard, West Bethesda, MD 20817-5700	**8. PERFORMING ORGANIZATION REPORT NUMBER** NSWCCD-61-TR-2015/31
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Naval Surface Warfare Center, Carderock Division 9500 MacArthur Boulevard West Bethesda, MD 20817-5700	**10. SPONSOR/MONITOR'S ACRONYM(S)** **11. SPONSOR/MONITOR'S REPORT NUMBER(S)** N/A

12. DISTRIBUTION / AVAILABILITY STATEMENT

Distribution Statement A: Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT:

A new computational procedure for including welding-induced residual stresses and distortions from weld simulations in the SYSWELD software code in structural Finite Element Analysis (FEA) simulations performed in the Abaqus FEA code is presented. The translation of these results is accomplished using a newly developed Python script. Full details of extracting the results from SYSWELD, translating these results using the newly developed Python script, and including the results in Abaqus are provided through an illustrative example analysis of a butt-welded aluminum plate. The development of this capability is motivated by the Navy's increased use of lightweight aluminum structures where including the effects of welding in structural components can be critical to understanding structural response. The integration of SYSWELD and Abaqus enables Code 60 engineers to increase the efficiency of design through the integrated computational materials engineering (ICME) paradigm, which emphasizes the use of computational tools to understand design performance, while minimizing the amount of traditional physical testing. This project is part of a program entitled "Integrated Computational Design and Analysis – Aluminum Ship Materials and Structural Performance (Al ICDA)."

15. SUBJECT TERMS

Computational simulation, SYSWELD, Abaqus, residual stress, distortion

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. RESPONSIBLE PERSON** Charles Fisher
a. REPORT UNCLASSIFIED	**b. ABSTRACT** UNCLASSIFIED	**c. THIS PAGE** UNCLASSIFIED		23	**19b. TELEPHONE NUMBER** (301) 227-4969

CONTENTS

Page

FIGURES

Page

ADMINISTRATIVE INFORMATION

The work described in this report was performed by the Welding, Processing, and Nondestructive Evaluation Branch (Code 611) and the Hull Response and Protection Branch (Code 664) of the Survivability, Structures, Materials, and Environmental Department of the Naval Surface Warfare Center, Carderock Division (NSWCCD). The work was funded in FY14 by Section 219 internal research funds under a project entitled “Integrated Computational Design and Analysis – Aluminum Ship Materials and Structural Performance” and was overseen by Jack Price, Director of Research at NSWCCD. This cross-division program was initiated to enable the development of a seamless computational capability to perform calculations from welding through structural response and failure.

ACKNOWLEDGEMENTS

The authors would like to thank Maria Posada and Matthew Sinfield (NSWCCD Code 611) for their technical assistance in the completion of this work. In addition, Brian Shula and Chandrashekhar Kanetkar, Senior Applications Engineers in the Welding Division at ESI Group – North America, served as external collaborators for the computational modeling effort.

This page intentionally left blank

EXECUTIVE SUMMARY

A new computational procedure for including welding-induced residual stresses and distortions from weld simulations in the SYSWELD software code in structural Finite Element Analysis (FEA) simulations performed in the Abaqus FEA code is presented. The translation of these results is accomplished using a newly developed Python script. Full details of extracting the results from SYSWELD, translating these results using the newly developed Python script, and including the results in Abaqus are provided through an illustrative example analysis of a butt-welded aluminum plate. The development of this capability is motivated by the Navy's increased use of lightweight aluminum structures where including the effects of welding in structural components can be critical to understanding structural response. The integration of SYSWELD and Abaqus enables Code 60 engineers to increase the efficiency of design through the integrated computational materials engineering (ICME) paradigm, which emphasizes the use of computational tools to understand design performance, while minimizing the amount of traditional physical testing. This project is part of a program entitled "Integrated Computational Design and Analysis – Aluminum Ship Materials and Structural Performance (Al ICDA)."

BACKGROUND

As a result of increasing requirements for high-speed vessels, the U.S. Navy is increasing use of lightweight structures, most notably in aluminum-based construction, where welding-induced stresses and distortions can have a significant impact on structural behavior. The effect of these welding-induced quantities often manifest themselves on a structural level, such as crack development with no appreciable global structural load as a result of residual stresses or premature buckling under compressive loading. In combination with this, many of these high-speed structures exhibit materials, welding configurations, and structural details that differ from historical U.S. Navy structures. As such, it is difficult to perform evaluations or improve the design of these structures using well-developed "rule of thumb" criteria.

Since a full testing program examining the performance of each detail under typical naval shiploads is not economically feasible, extensive computations accompanied by targeted testing are required for evaluation and development of new designs. However, in order to perform such computations, an efficient method to include welding-induced stresses and initial distortions in these computations is required. This approach, falling within the Integrated Computational Materials Engineering (ICME) paradigm, hinges on both validation and verification (V&V) of the applied methods and the seamless communication of the computational tools of interest.

Historically, computational tools for analyzing welding and structural behavior have been used in an isolated setting. In the present effort, a process to transfer results from weld process simulations, performed using the SYSWELD welding simulation Finite Element Analysis (FEA) software package, to coupon and structural scale calculations, performed in the widely-utilized Abaqus FE software package, is described. While Abaqus has been used extensively for simulating the mechanical behavior of aluminum structural components [1-4], no documented integration of SYSWELD and Abaqus was identified.

Overview of SYSWELD Software

SYSWELD is a commercially-available FEA software package developed by ESI Group for use in thermo-mechanical and thermo-metallurgical welding process simulations. Specific welding processes can be directly simulated with the ability to input welding parameters, including travel speed, arc voltage/current, weld pool size, and clamping conditions. SYSWELD uses material databases that contain the thermal, metallurgical, and mechanical material properties for the materials being simulated. Though widely used and accepted for capturing the welding process in steel, there are relatively few examples of SYSWELD used to simulate aluminum welding processes [5-7]. The reader is referred to the SYSWELD Toolbox for full details on the usage and input format for SYSWELD software [8].

Overview of Abaqus Software

Abaqus FEA is a widely utilized and commercially-available, general-purpose FE software package developed by Dassault Systèmes. The software enables modeling of materials, components, and assemblies under different loading conditions so that stress, strain, and other relevant quantities can be determined. Abaqus analyses readily include pre-existing material/structural states such as initial residual stresses and distortions. The reader is referred to Abaqus FEA documentation for full details on the usage and input format for Abaqus software [9].

PROCEDURE

The transfer of results from SYSWELD to Abaqus FEA is achieved using the post-processing tools supplied with SYSWELD, along with available Abaqus input keywords. A newly-developed script**, `SYSWELD_ABQ_field.py`,** performs this transfer (see **Appendix** for a full code listing). A walk-through of the process is given below. A description of the process for generating the necessary files from SYSWELD is described followed by instructions of using the script and including the translated SYSWELD results in an Abaqus analysis.

The transfer of SYSWELD results to Abaqus involves several steps in both software tools:

1. Generate output data of the field quantities of interest, namely stress and displacement, using the SYSWELD post-processing module.
2. Transfer mesh from SYSWELD to Abaqus using SYSWELD's **`*.inp`** mesh output capability. Alternatively, SYSWELD can read in the starting Abaqus mesh. It is critical that elemental and nodal numbering be preserved and that any mesh generated in Abaqus be in the "flat" format; i.e., not utilize parts and assemblies. Alternatively, SYSWELD analyses can be performed using a mesh imported from Abaqus.
3. Translate SYSWELD field quantities to Abaqus-interpretable format using **`SYSWELD_ABQ_field`** script.

4. Run subsequent Abaqus FEA analysis with `*INCLUDE` keyword to include initial residual stress and distortion input files generated by `SYSWELD_ABQ_field` script.

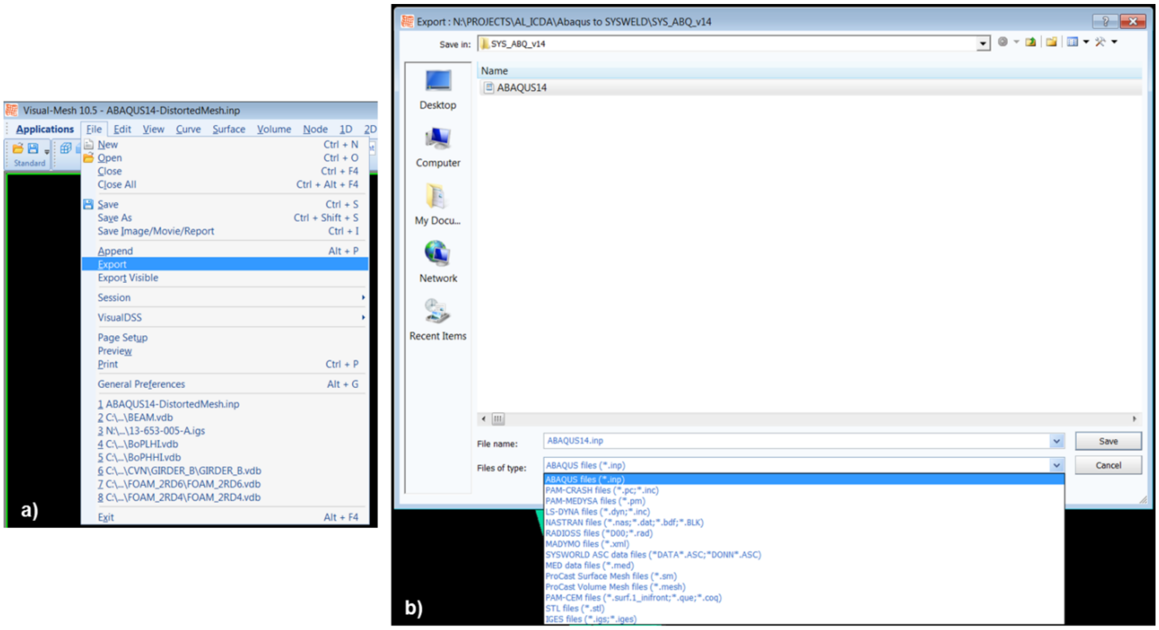
   A full description of these steps is provided below.

SYSWELD Results Preparation

The output required for the translation script consists of a series of files tabulating each stress component (XX, YY, ZZ, XY, XZ, YZ) for every element in the form `STRESSES_ELE_XX.TXT`**,** each distortion component (X, Y, Z) in the form `DISTORTION_NOD_X.TXT`, and, for most cases, a valid exported Abaqus mesh. The steps described below document how to generate the necessary output text files from a SYSWELD simulation using the graphical interface of Visual Environment 10.5 (including Visual Mesh, Visual Weld, and Visual Viewer). The simulation results examined were generated using SYSWELD 2015.

SYSWELD Mesh Output

The mesh from the original CAD file may have been modified during the weld modeling process within SYSWELD. Therefore, after the simulation, open the `<file name>.vdb` file within the Visual Mesh program of SYSWELD. To export the mesh, go to the 'File' menu and click on 'Export' (see **Figure 1a**). This brings up a new window (see **Figure 1b**) enabling the user to save the mesh in numerous file formats; in this case, 'ABAQUS files (*.inp)' should be selected.


Visual-Mesh 10.5 - ABAQUS14-DistortedMesh.inp
Applications
File
Edit
View
Curve
Surface
Volume
Node
1D
2D
Standard
New Ctrl + N
Open Ctrl + O
Close Ctrl + F4
Close All Ctrl + Alt + F4
Save Ctrl + S
Save As Ctrl + Shift + S
Save Image/Movie/Report Ctrl + I
Append Alt + P
Export
Export Visible
Session
VisualDSS
Page Setup
Preview
Print Ctrl + P
General Preferences Alt + G
1 ABAQUS14-DistortedMesh.inp
2 C:\...\BEAM.vdb
3 N:\...\13-653-005-A.igs
4 C:\...\BoPLHI.vdb
5 C:\...\BoPHHI.vdb
6 C:\...\CVN\GIRDER_B\GIRDER_B.vdb
7 C:\...\FOAM_2RD6\FOAM_2RD6.vdb
8 C:\...\FOAM_2RD4\FOAM_2RD4.vdb
Exit Alt + F4
a)
Export : N:\PROJECTS\AL_ICDA\Abaqus to SYSWELD\SYS_ABQ_v14
Desktop
Computer
My Docu...
Network
Recent Items
Name
ABAQUS14
Save
Cancel
ABAQUS files (*.inp)
PAM-CRASH files (*.pc;*.inc)
PAM-MEDYSA files (*.pm)
LS-DYNA files (*.dyn;*.inc)
NASTRAN files (*.nas;*.dat;*.bdf;*.BLK)
RADIOSS files (*D00;*.rad)
MADYMO files (*.xml)
SYSWORLD ASC data files (*DATA*.ASC;*DONN*.ASC)
MED data files (*.med)
ProCast Surface Mesh files (*.sm)
ProCast Volume Mesh files (*.mesh)
PAM-CEM files (*.surf.1_inifront;*.que;*.coq)
STL files (*.stl)
IGES files (*.igs;*.iges)
b)

Save in: SYS_ABQ_v14

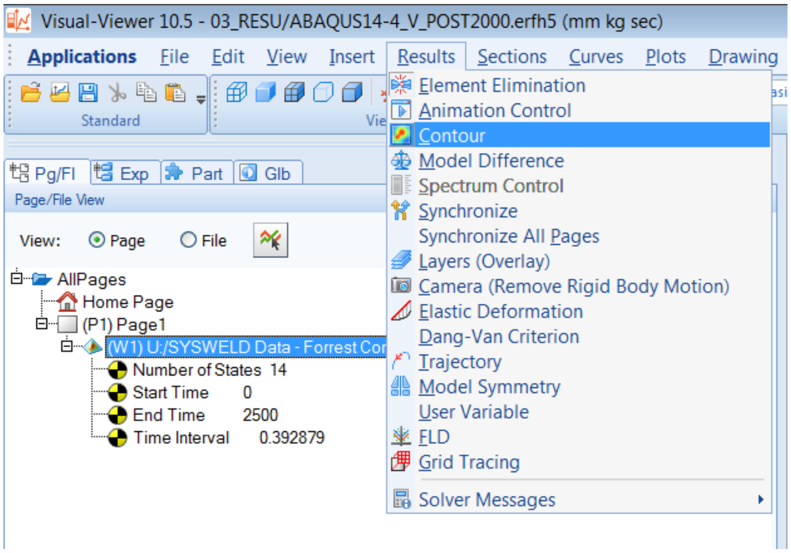
File name: ABAQUS14.inp

Files of type: ABAQUS files (*.inp)

Figure 1. Screen shots of the Visual Mesh function within SYSWELD showing how to **a)** export mesh and **b)** save to the appropriate Abaqus file format.

Distortion Output

The distortion of the mesh after the SYSWELD simulation can be viewed by opening the `<file name>_V_POST2000.fdb` file within the Visual Viewer program of SYSWELD. To view the distortion, go to the 'Results' menu and click on 'Contour' (see **Figure 2**). This brings up a new window (see **Figure 3**). Depending on settings, the user must click differently to obtain the same resulting views. Under the 'Tree' menu, if set to *Quantity*, open the 'Kinematics' drop down menu, and set the 'Displacement' drop down menu to either *X*, *Y*, or *Z* based on the principle nodal direction under investigation (see **Figure 3a**). Under the 'Tree' menu, if set to *Entity*, open the 'Node' drop down menu, and set the 'Displacement' drop down menu to either *X*, *Y*, or *Z* based on the principle nodal direction under investigation (see **Figure 3b**).



Number of States 14
Start Time 0
End Time 2500
Time Interval 0.392879

Figure 2. Screen shot of the Visual Viewer function within SYSWELD showing how to view the 'Contour' window used to display distortion and stress levels among other predicted properties.

The distortion results can be output in text format by clicking to the final step of the simulation. Next, go to the 'File' menu and click 'Save As → Ascii Contour' (see **Figure 4a**). This brings up a new window (see **Figure 4b**). Under the 'Entity' menu, ensure the text is set to *NODE*. Finally, click `DISPLACEMENTS_NOD_X` (or `Y` or `Z`), and save the output data to an appropriate location.

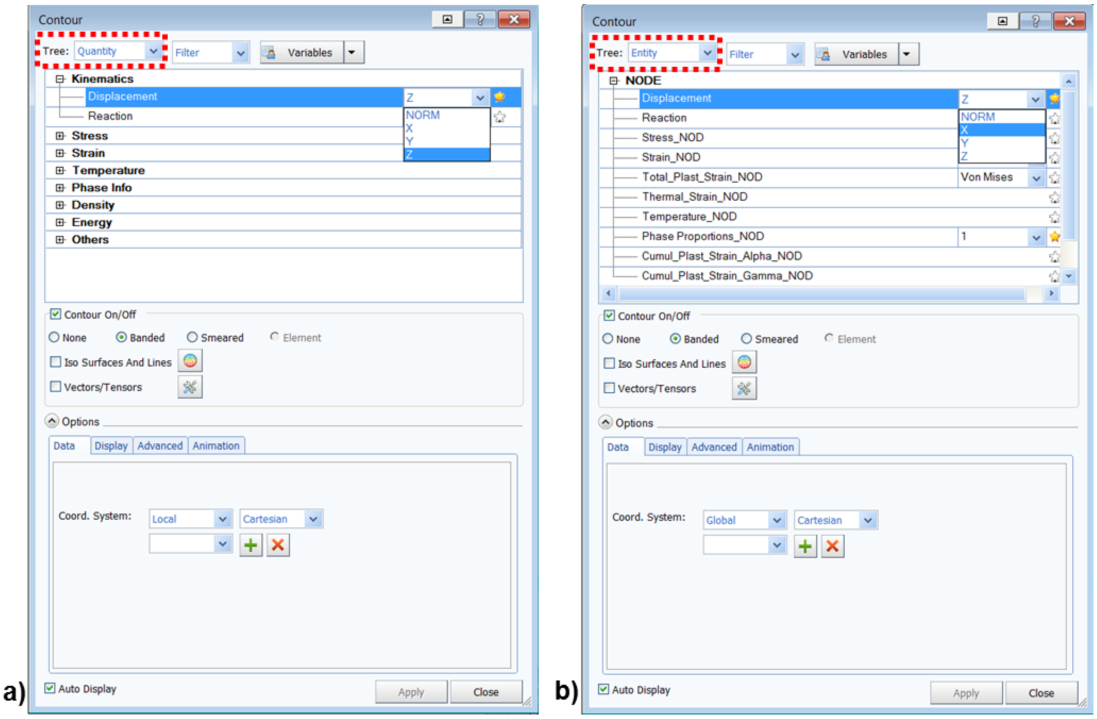


Figure 3. Screen shots of the 'Contour' window showing how to display the mesh displacement depending on if the 'Tree' menu, highlighted with dotted red boxes, is set to **a)** *Quantity* or **b)** *Entity*.

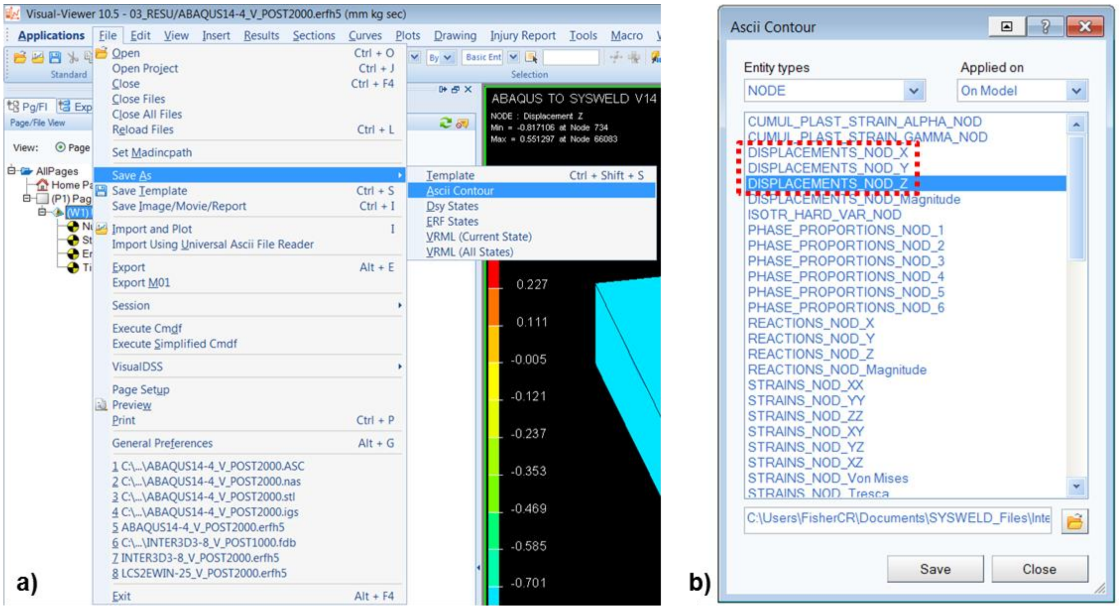
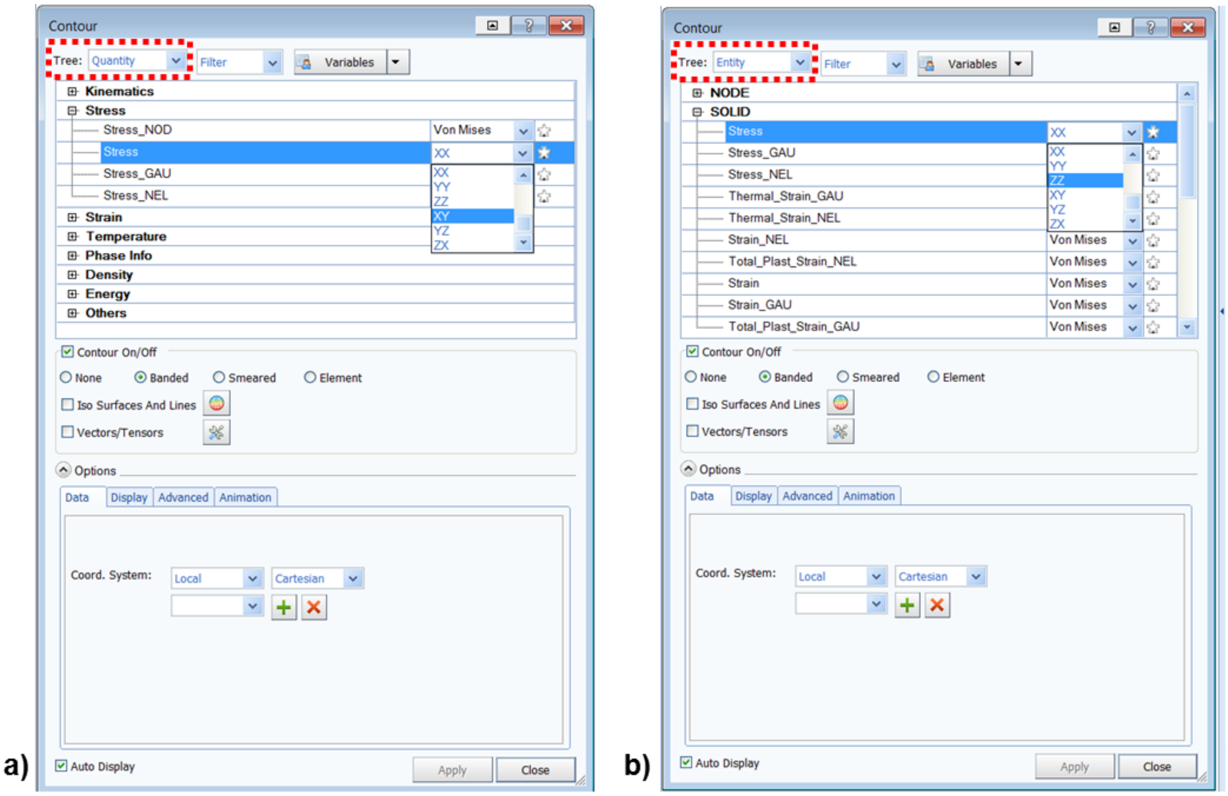


Figure 4. Screen shots of the Visual Viewer function showing **a)** drop-down menus to save an ASCII contour (text-based file) and **b)** different properties available to be saved as such. For this case, `DISPLACEMENTS_NOD_X` (or `Y` or `Z`), as highlighted by the dotted red box, are required.

Stress Output

The predicted stress from the welding simulation can be viewed by opening the `<file name>_V_POST2000.fdb` file within the Visual Viewer program of SYSWELD. Similar to displaying the nodal distortion, to view the stress levels go to the ‘Results’ menu and click on ‘Contour’ (as in **Figure 2**) which brings up a new window. Depending on settings, the user must click differently to obtain the same resulting views. Under the ‘Tree’ menu, if set to *Quantity*, open the ‘Stress’ drop down menu, and set the second ‘Stress’ drop down menu to *XX*, *YY*, *ZZ*, *XZ*, *YZ*, or *XY* based on the principle stress direction under investigation (see **Figure 5a**). Under the ‘Tree’ menu, if set to *Entity*, open the ‘Solid’ drop down menu, and set the ‘Stress’ drop down menu to *XX*, *YY*, *ZZ*, *XZ*, *YZ*, or *XY* based on the principle stress direction under investigation (see **Figure 5b**).

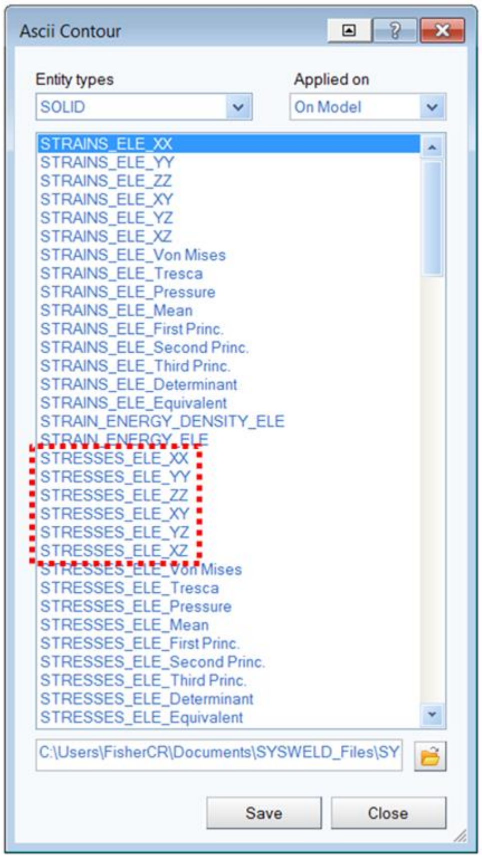


a)
Tree: Quantity
Stress_NOD: Von Mises
Stress: XX
☑ Contour On/Off
○ None ● Banded ○ Smeared ○ Element
☐ Iso Surfaces And Lines
☐ Vectors/Tensors
Coord. System: Local Cartesian
☑ Auto Display

b)
Tree: Entity
Stress: XX
Strain_NEL: Von Mises
Total_Plast_Strain_NEL: Von Mises
Strain: Von Mises
Strain_GAU: Von Mises
Total_Plast_Strain_GAU: Von Mises
☑ Contour On/Off
○ None ● Banded ○ Smeared ○ Element
☐ Iso Surfaces And Lines
☐ Vectors/Tensors
Coord. System: Local Cartesian
☑ Auto Display

Figure 5. Screen shots of the ‘Contour’ window showing how to display the predicted stress depending on if the ‘Tree’ menu, highlighted with dotted red boxes, is set to either **a)** *Quantity* or **b)** *Entity*.

The stress results can be output in text format by clicking to the final step of the simulation. Next, go to the ‘File’ menu and click ‘Save As → Ascii Contour’ (see **Figure 4a**); this is the same as for distortion output. For stress, however, in the new window (see **Figure 6**) which opens, ensure the text is set to *SOLID* under the ‘Entity’ menu. Finally, click `STRESSES_ELE_XX` (or `YY`, `ZZ`, `XZ`, `YZ`, or `XY`), and save the output data to an appropriate location.



Entity types: SOLID

Applied on: On Model

C:\Users\FisherCR\Documents\SYSWELD_Files\SY

Figure 6. Screen shot of the Visual Viewer function showing the different properties available to be saved as an ASCII contour (text-based file). To save stress data, `STRESSES_ELE_XX` (or `YY`, `ZZ`, `XZ`, `YZ`, or `XY`), as highlighted by the dotted red box, are required.

Running Translation Script

Once the stress and distortion output is generated from SYSWELD, the **`SYSWELD_ABQ_field.py`** may be run. All SYSWELD output must be in the same directory. The script itself can be run either using Abaqus' built-in Python shell or any external installation of Python that includes the **`csv`** and **`argparse`** modules. The script was tested to work in Abaqus 6.14 with no additional software beyond a valid Abaqus FEA installation.

The **`SYSWELD_ABQ_field.py`** script has separate functionalities for nodal and element integration point quantities and must be executed twice: once for stress and once for distortion. The stress and distortion quantities are handled as follows:

- Residual stresses, described at element integration points, are processed and formatted using the **`*INITIAL CONDITIONS, TYPE=STRESS`** Abaqus keyword. SYSWELD generates an individual file for each component of stress $(\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{xz}, \sigma_{yz})$. The script generates a standalone Abaqus input file, **`stress.inp`**, that provides the individual stress components in a six-column vector that is tabulated for each element.

- Initial distortions or displacements, a nodal quantity with three components (X, Y, Z) are converted to the **`*IMPERFECTION, INPUT=FILENAME`** Abaqus keyword format. SYSWELD generated an individual file for each component of displacement $(\Delta x, \Delta y, \Delta z)$. A file **`displacement.csv`** is generated that assembles these individual displacement components into a three-column vector associated with each node number.

The **`SYSWELD_ABQ_field.py`** script is invoked through a command-line driven interface using the syntax:

```
abaqus python SYSWELD_ABQ_field.py -fieldType <displacement|stress>
-path <path to SYSWELD field data> -scaleFactor <SF>
```

The arguments describe:

- **`fieldType`**: Field of interest. Stresses are six-value tensor quantities at each element whereas initial displacements are three-value vectors at each node.
- **`path`**: Path to Sysweld-generated stress or displacement data.
- **`scaleFactor`**: Scale factor to convert SYSWELD units to units in Abaqus analysis for field of interest.

A help description can be obtained by executing:

```
abaqus python SYSWELD_ABQ_field.py –h
```

The commands:

```
python SYSWELD_ABQ_field.py -fieldType stress –path
/myDirectory/results -scaleFactor 1e6

python SYSWELD_ABQ_field.py -fieldType displacement –path
/myDirectory/results -scaleFactor 0.001
```

will generate **`stress.inp`** and ***`displacement.csv`*** files using the SYSWELD stress and displacement data located in **`/myDirectory/results`**. A multiplier of 1e6 and 0.001 will be applied to all stress values and displacement values, respectively. It is the user’s responsibility to ensure that consistent unit systems are maintained. If a component is not defined by the data provided from SYSWELD, a null value is assumed to ensure either a three-column vector for displacement or a six-column stress vector for stress is obtained.

It is strongly recommended that a static step be undertaken with the SYSWELD-generated residual stresses and distortions to ensure precise equilibrium. Once this step is taken, the Abaqus analysis of interest can be performed.

USAGE EXAMPLE

The transfer of SYSWELD results to Abaqus was demonstrated for a B2V.1 joint, which is a single-V butt joint without a backing bar [10]. The simulated material was Al 5052 using the gas-metal arc welding (GMAW) process. A simulation of the welding process of this joint was conducted in SYSWELD and is documented in another NSWCCD report [11]. While this specific weldment does not represent an actual shipboard application, it was selected because it demonstrated numerous weld processing conditions, including multiple weld passes, flipping the weldment, and a back-gouge, in order to validate the SYSWELD to Abaqus technique. **Figure 7** shows a view of the mesh, clamping conditions, and the three weld passes, shown in purple, pink, and gold coloring.

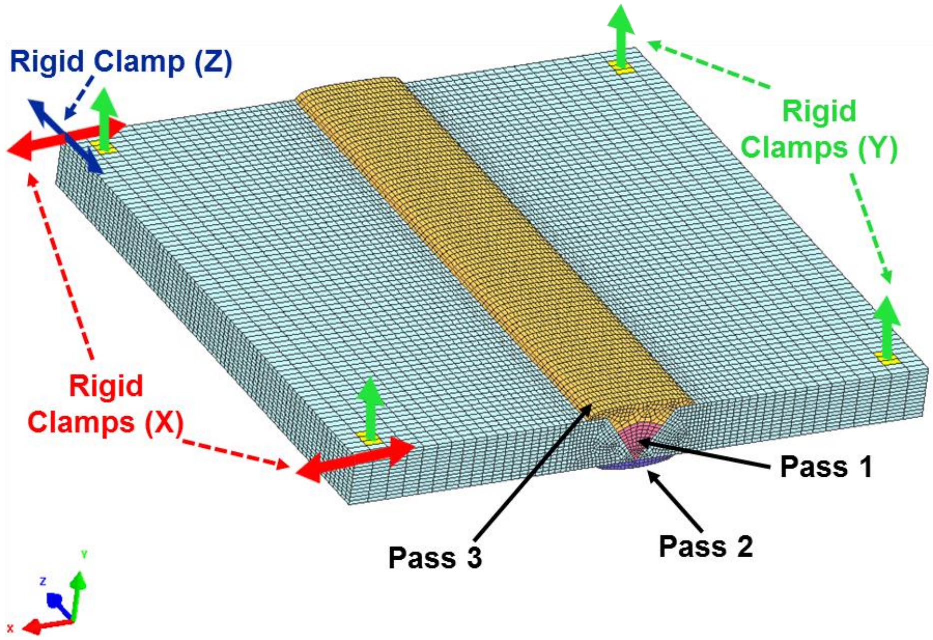


Figure 7. SYSWELD-generated model of an aluminum B2V.1 joint showing the initial mesh and clamping conditions.

Translating Results

Once the SYSWELD output steps are complete, the **`SYSWELD_ABQ_field.py`** script is run to generate a stress.inp file. This file is directly included in the Abaqus analysis including the following keyword in the main input file:

```
*Include, input=stress.inp
```

A contour plot showing the final residual stress state from SYSWELD simulations and the initial state of an Abaqus analysis using these stresses is show in **Figure 8**. Note that the mesh was generated in SYSWELD, exported to Abaqus, and scaled from mm-kg-MPa to MKS units. The stresses were translated accordingly. Prior to undertaking further Abaqus analysis, it is strongly recommended that a static equilibrium step be performed to ensure exact static equilibrium.

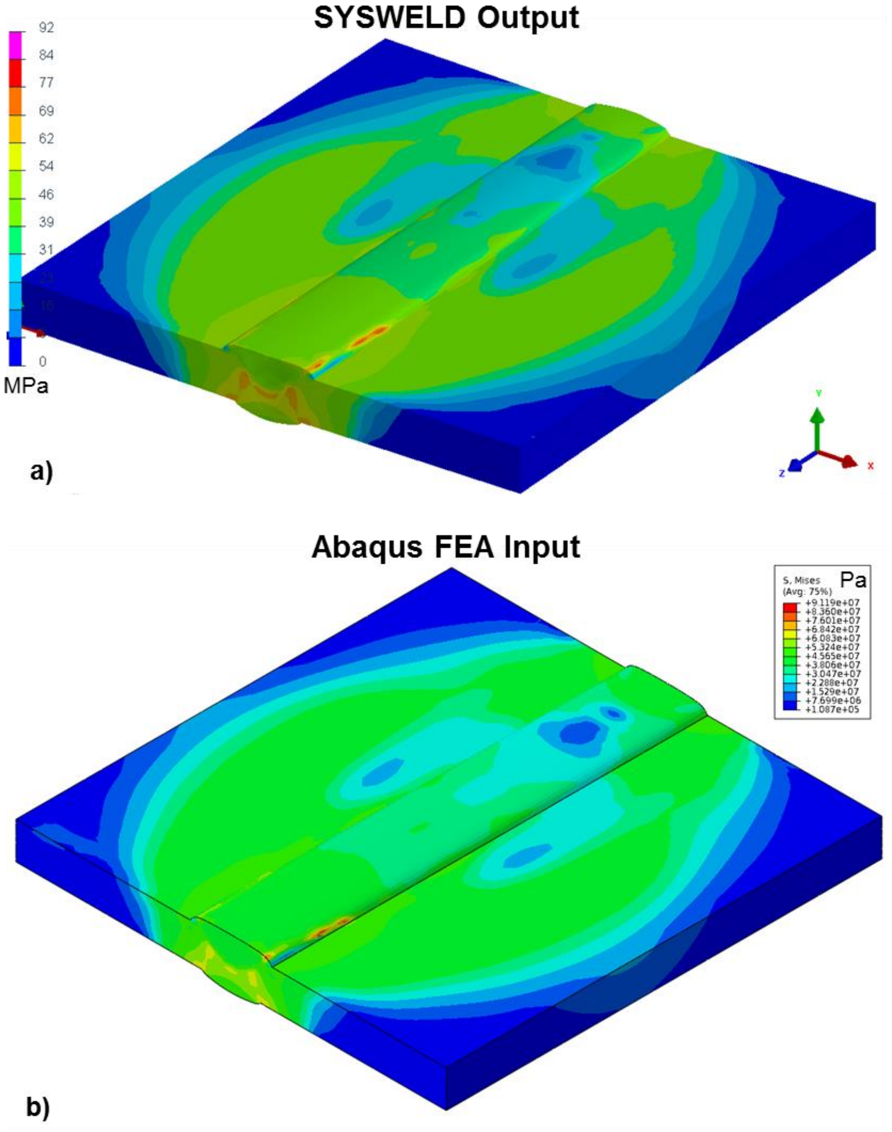


Figure 8. Comparison of Von Mises stress levels from **a)** SYSWELD simulation output values and **b)** same values translated to Abaqus FEA. Minor color variations are due to differences in contour coloring.

Using Initialized Local Weld Model in Structural FEA

With the ability to include initial residual stress and distortion from SYSWELD simulations in Abaqus structural analyses, global loads can readily be transferred into the local weld detail using the sub-modeling capability available in Abaqus FEA. This capability automatically maps the edge displacements of the coarse global mode to edge boundary conditions on the local model.

An example application of this is shown in **Figure 9**. The initial state of the local weld and the unloaded state are shown in **Figure 9a**. As displacement is applied to the edges of the global structural model, a complex stress state in the weld is observed to develop in **Figure 9b**.

This result is very different from the result obtained without including residual stresses from welding due to the high level of these stresses relative to the stress induced by structural loading.

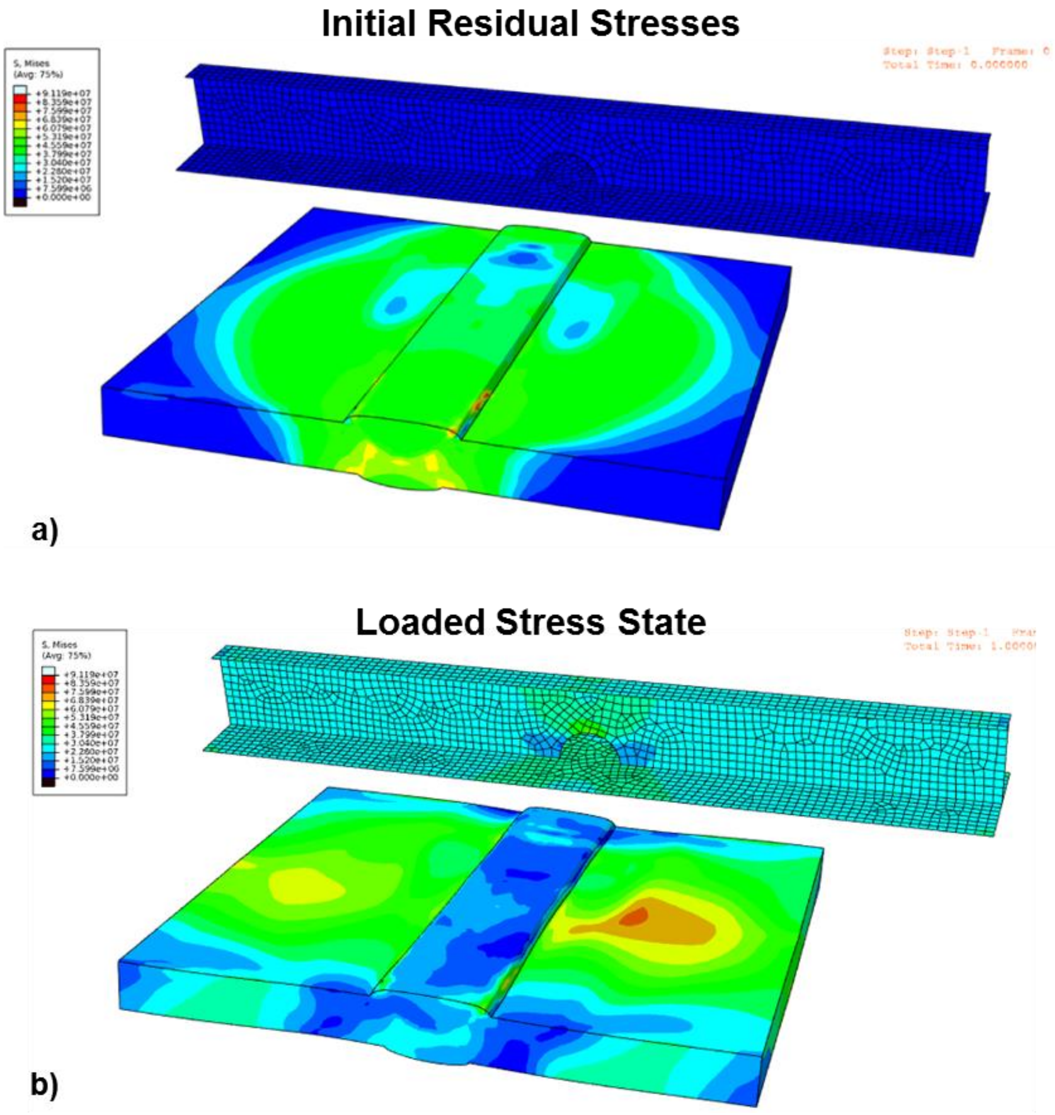


Figure 9. Embedded weld with initial residual stresses, determined by SYSWELD, in global shell mesh **a)** prior to a loading simulation in Abaqus, and **b)** post-loading in Abaqus. Notice the stresses in the weld and global shell mesh under uniaxial tension loading along stiffener direction. Coupling from global shell mesh to local weld mesh is fully automated using one-way.

CONCLUSIONS

A process and necessary software for translating SYSWELD results from inclusion in Abaqus FEA structural analyses has been developed and its usage demonstrated the successful integration of the two software programs. The application of this newly-developed capability to structural analysis of a multi-pass aluminum weld illustrates the inability of structural FEA calculations to capture local weld detail response. As failures often initiate at these locations, the U.S. Navy is now able to investigate how the welding process modifies the resultant fracture and fatigue behavior for structural components on its vessels.

REFERENCES

1. A. Belegundu, *Shape Optimization of Plates to Mitigate the Effects of Air Blast Loading*, U.S. Army Research Office Final Report, 50490-EG.1, June 2009.
2. I. Scheider, M. Schödel, W. Brocks, and W. Schönfeld, *Crack Propagation Analyses with CTOA and Cohesive Model: Comparison and Experimental Validation*, Engineering Fracture Mechanics, 2006, Vol. 73, pg. 252-263.
3. J. Zhou, M. Hayden, and X. Gao, *An Investigation of the Strain Rate and Temperature Effects on the Plastic Flow Stress and Ductile Failure Strain of Aluminum Alloys 5083-H116, 6082-T6, and a 5183 Weld Metal*, Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 2013, Vol. 227, No. 5, pg. 883-895.
4. Xiaosheng Gao, Tingting Zhang, Matthew Hayden, and Charles Roe, *Effects of the Stress State on Plasticity and Ductile Failure of an Aluminum 5083 Alloy*, International Journal of Plasticity, 2009, Vol. 25, Issue 12, pg. 2366-2382.
5. J.D. Francis, *Welding Simulations of Aluminum Alloy Joints by Finite Element Analysis*, Master's Thesis at Virginia Polytechnic Institute and State University, Blacksburg, VA, 2002.
6. M. Zain-ul-abdein, D. Nelias, J.-F. Jullien, F. Boitout, L. Dischert, and X. Noe, *Finite Element Analysis of Metallurgical Phase Transformations in AA 6056-T4 and Their Effects Upon the Residual Stress and Distortion States of a Laser-Welded T-Joint*, International Journal of Pressure Vessels and Piping, 2011, Vol. 88, pg. 45-56.
7. Z.H. Guo, X.Y. Ou, G.W. Shuai, and Y.H. Chen, *Numerical Simulation of Temperature Field for TIG Welding of Aluminum Alloy Sheet Based on SYSWELD*, Advanced Materials Research, 2012, Vol. 472-475, pg. 1945-1949.
8. SYSWELD Toolbox 2015, *Weld and Heat Treatment Simulation Manual*, ESI Group, Farmington Hills, MI, USA.
9. ABAQUS (2015), *ABAQUS Documentation*, Dassault Systèmes, Providence, RI, USA.
10. MIL-STD-22D, Department of Defense Design Criteria: Welded Joint Design, 29 August 1979.
11. C.R. Fisher and K. Nahshon, *SYSWELD Computational Modeling for Data Transfer to Abaqus FEA Software*, NSWCCD-61-TR-2015/32, November 2015.

APPENDIX: SYSWELD_ABQ_field.py CODE

```
# SYSWELD_ABQ_field.py
# Written by Ken Nahshon (ken.nahshon@navy.mil), Naval Surface Warfare Center
# Carderock Division
# 27 July 2015
# This code is subject to all USC regulations. NSWCCD shall not be held liable for any use or
# misuse of this code. The code is stictly provided as-is.
# ------------------------------------------------------------------------------------------------
# Converts field quantities from SYSWELD to ABAQUS
# Input consists of a directory with SYSWELD results e.g. DISPLACEMENTS_NOD_X.txt etc and STRESSES_ELE_XX.txt
# Output consists of a file stress.inp consisting of:
#
#         stress.inp: *INITIAL CONDITION pre-stress values. The stress file
#           can be included using *Include, input=stress.inp
#           or directly copied and pasted into the ABAQUS input file.
#
#         displacement.csv: delta x, y, and z for moving nodal coordinates.
#           ABAQUS can move the nodal coordinates
#           using *IMPERFECTION input=displacement.csv
# ------------------------------------------------------------------------------------------------
# Usage:
#
# python SYSWELD_ABQ_field.py -fieldType <field_type> -path <path to SYSWELD files> -scaleFactor <scaleFactor>
#
# Where,
#    field_type can be either 'stress' or 'displacement'. Currently, other fields are not supported
#    scaleFactor is a floating point value multiplying the SYSWELD results to account for unit changes
#    working_directory is the location of SYSWELD output files
#
# Example:
#
# python SYSWELD_ABQ_field.py -fieldType stress -scaleFactor 1e6 -path /runs/weldcalc/
#
# Will execute the script to generate a stress.inp file in /runs/weldcalc/ using the SYSWELD output located
# in that directory with stresses from SYSWELD multiplied by the scale factor 1e6
#
# ------------------------------------------------------------------------------------------------

import os
import csv
import sys
import argparse

def filesWithString(fileStr, directory):
    """Returns a list of filenames with fileStr in name in directory\n
    Usage: filesWithString(fileStr, directory)"""
    listing=os.listdir(directory)
    return [elem for elem in listing if elem.find(fileStr)>0]

def openInputFile(inputFileName, readWriteOption):
    """Safe file open with exceptions"""
    import sys
    print 'Processing: ', inputFileName
    try:
        inputFile=open(inputFileName, readWriteOption)
        return inputFile
    except IOError as e:
        print "Error in file open in openInputFile:\n"
        print "I/O error({0}): {1}".format(e.errno, e.strerror)
    except:
        print "Unexpected error:", sys.exc_info()[0]
        raise

def openCsv(fileName, commentChar = ('#', '!', '%'), **kwargs):
    """
    Read a .csv file with fileName (string) and return a list of lines where each line is a list
    of entries seperated by commas in fileName. Routine strips out lines beginning with
    commentChars and additional whitespace. Additional comment markers can be included.
    Usage: \n\n
    parseCSVtoList(fileName, commentChar = ('#', '!', '%'))
    """
    whitespace=(' ', '\t')                  # Define whitespace to ignore
    f=openInputFile(fileName, 'r')
    reader=csv.reader(f, **kwargs)
    lineCount=0
    inputData=[]
    for row in reader:
        if (len(row) == 0) or (row[0][0] in commentChar): # Check for comment characters and skip entire line
            continue       # Skip empty lines
        else:
            # Eliminate additional whitespaces
            for char in whitespace:
                row=[row[index].replace(char, "") for index in range(0, len(row))]
            inputData.append(row)
        lineCount += 1
    return inputData

# ********************************************************************************
# MAIN CODE
# ********************************************************************************

# Show title info

titleBlock = """\n*************************************************************************
             \nSYSWELD_ABQ_field.py, a tool to convert SYSWELD fields to ABAQUS formats.
             Currently supports stress and displacment types.
             \n*************************************************************************"""

#print titleBlock

parser = argparse.ArgumentParser(description=titleBlock)
parser.add_argument('-fieldType', required=True, metavar=('<displacement|stress>'), type=str, nargs=1,
help='Specify field type')
parser.add_argument('-path', required=True, metavar=('<Path to SYSWELD field data>'), type=str, nargs=1,
help='Specify directory for SYSWELD files')
parser.add_argument('-scaleFactor', required=True, metavar=('<SF>'), type=float, nargs=1, help='Specify scale
factor for field')
parser._optionals.title = "Flag arguments"
# Parse arguments
args = parser.parse_args()

# Get command line input
fieldType = args.fieldType[0]
scaleFactor = args.scaleFactor[0]
directory = args.path[0]

# Make sure directory exists
if os.path.exists(directory) is False:
    print '\n*** Fatal Error, %s is not a valid path ***' %directory
    sys.exit(0)

# Make sure / is last character in directory name
#if directory[-1] != '/':
#    directory = directory + '/'

# Precalculation output

print '\nField type: %s' %fieldType
print 'Multiplying SYSWELD %s by scale factor: %s' %(fieldType, scaleFactor)
print 'Working directory: %s\n' %directory

# Switch to working directory
os.chdir(directory)

# --------------------------
# Set up valid field types, directions
# --------------------------
#
# Valid field types
fieldList = ['displacement', 'stress']
# List of directions for tensors (elements)
directionList=['XX', 'YY', 'ZZ', 'XY', 'YZ', 'ZX']
# List of directions for nodes
nodalDirectionList=['X', 'Y', 'Z']

# Check for supported field type, set file prefix for SYSWELD. If not supported, exit.
if fieldType == 'stress':
    prefix = 'STRESSES_ELE_'
elif fieldType == 'displacement':
    prefix = 'DISPLACEMENTS_NOD_'
else:
    print '** ERROR: following field type is not supported: ', fieldType
    sys.exit()

# Perform actions based on field type

# Stress translation
if fieldType == 'stress':
    # Set a dictionary of stress values, read stresses from direction if file exists
    stress = dict()
    for direction in directionList:
        fileName = prefix + direction + '.txt'
        if fileName in os.listdir(directory):
            data=openCsv(directory+fileName, delimiter='\t', skipinitialspace = 'True')
            # Strip header off data file. Currently, SYSWELD fills first 8 lines
            data=data[8:]
            # Read element list
            elList = [ int(dataLine[0]) for dataLine in data]
            # Read stresses, break into directions using dictionary
            stress[direction] =  [float(dataLine[1])*scaleFactor for dataLine in data]
            numElements = len(elList)
    # Backfill with zeroes for stress components that don't exist
    for direction in directionList:
        if direction not in stress.keys():
            stress[direction] =  [0.0 for line in range(0, numElements)]

    # write out data as six column vector

    with open(directory + 'stress.inp', 'wt') as f:
        f.write('**Autogenerated prestress from SYSWELD\n')
        f.write('*Initial Conditions, type=' + fieldType  + '\n')
        for i in range(0,numElements):
            line = '{0}, {1}, {2}, {3}, {4}, {5}, {6}\n'.format(str(elList[i]),
                str(stress['XX'][i]), str(stress['YY'][i]), str(stress['ZZ'][i]),
                    str(stress['XY'][i]), str(stress['YZ'][i]), str(stress['ZX'][i]))
            f.write(line)
    print '\nWrote %s in directory %s\n' %('stress.inp', directory)

if fieldType == 'displacement':
    displacement = dict()
    nodeList = list()
    for direction in nodalDirectionList:
        fileName = prefix + direction + '.txt'
        if fileName in os.listdir(directory):
            data=openCsv(directory+fileName, delimiter='\t', skipinitialspace = 'True')
            # Strip header off data file
            data=data[8:]
            nodeList = [ int(dataLine[0]) for dataLine in data]
            displacement[direction] =  [float(dataLine[1])*scaleFactor for dataLine in data]

    numNodes = len(nodeList)
    # Backfill with zeroes
    for direction in nodalDirectionList:
        if direction not in displacement.keys():
            displacement[direction] =  [0.0 for line in range(0, numNodes)]

    # write out data

    with open(directory + 'displacement.csv', 'wt') as f:
        f.write('**Autogenerated initial distortions from SYSWELD\n')
#        f.write('*Initial Conditions, type=' + fieldType  + '\n')
        for i in range(0,numNodes):
            line = '{0}, {1}, {2}, {3}\n'.format(str(nodeList[i]), str(displacement['X'][i]),
str(displacement['Y'][i]), str(displacement['Z'][i]))
            f.write(line)
    print '\nWrote %s in directory %s\n' %('displacement.csv', directory)
```

This page intentionally left blank

DISTRIBUTION

EXTERNAL

Copies

DEFENSE TECHNICAL
INFORMATION CENTER 2
727 JOHN J KINGMAN ROAD
SUITE 0944
FORT BELVOIR, VA 22060-6218

COMMANDER
ATTN: SEA 05P2
NAVAL SEA SYSTEMS COMMAND 5
1333 ISAAC HULL AVENUE S.E.
WASHINGTON NAVY YARD
WASHINGTON, DC 20376
ATTN: Bjornson, Blackburn, McGrorey,
Novack, Zook

COMMANDER
ATTN: SEA 05P4
NAVAL SEA SYSTEMS COMMAND 2
1333 ISAAC HULL AVENUE S.E.
WASHINGTON NAVY YARD
WASHINGTON, DC 20376
ATTN: Gardner, Sensharma

COMMANDER
ATTN: SEA 05V
NAVAL SEA SYSTEMS COMMAND 2
1339 PATTERSON AVE
WASHINGTON NAVY YARD
WASHINGTON, DC 20376
ATTN: Jent, Perry

COMMANDER
ATTN: SEA 05D2
NAVAL SEA SYSTEMS COMMAND 2
1333 ISAAC HULL AVENUE S.E.
WASHINGTON NAVY YARD
WASHINGTON, DC 20376
ATTN: Izenson, Lawler

COMMANDER
ATTN: SEA 05D4
NAVAL SEA SYSTEMS COMMAND 1
1333 ISAAC HULL AVENUE S.E.
WASHINGTON NAVY YARD
WASHINGTON, DC 20376
ATTN: Earnest

EXTERNAL

Copies

OFFICE OF NAVAL RESEARCH 3
875 N RANDOLPH ST
ARLINGTON, VA 22217
ATTN: Barsoum, Hess, Mullins

NSWCCD INTERNAL DISTRIBUTION

Code	*Name*	*Copies*
60		1
61	DeLoach	1
611	Posada	1
611	Sinfield	1
611	Fisher	2
611	Tran	1
611	Scheck	1
612	Roe	1
612	Jones	1
612	Hayden	1
653	Grisso	1
653	Adler	1
653	Margelowsky	1
653	Nesson	1
653	Shilling	1
654	Amanuel	1
654	Dukes	1
664	Nahshon	2
614	(Report Documentation Page)	1
617	(Report Documentation Page)	1
63	(Report Documentation Page)	1
65	(Report Documentation Page)	1
66	(Report Documentation Page)	1
3442	(TIC)	1

NAVSEA

WARFARE CENTERS
Carderock